N88-11163

FLOW IN A MODEL TURBINE STATOR*

R.C. Buggeln, S.J. Shamroth and W.R. Briley
Scientific Research Associates, P.O. Box 498, Glastonbury, CT 06033

INTRODUCTION

A major problem area associated with the successful design and operation of modern gas turbine engines is the engine hot section. Of particular interest in the present study is the turbine. The turbine section represents a considerable challenge as it contains significant regions of complex three-dimensional flow including both aerodynamic and heat transfer phenomena. In particular, the turbine flow field contains several features which makes its analysis a formidable problem. These include complex geometry, multiple length scales, three-dimensional effects, possible strong secondary flows, possible flow separation at off-design operation, possible transonic effects, and possibly important unsteady effects.

In considering possible approaches to this problem, three approaches are evident. These are (i) inviscid analyses, (ii) inviscid analyses with boundary layer corrections, and (iii) full Navier-Stokes analyses. In view of the complex nature of the flow, the need to predict heat transfer and flow losses, the possible appearance of separation and strong secondary flows, etc., the present effort is focusing upon a Navier-Stokes approach to the three-dimensional turbine stator problem. The advantages of a full Navier-Stokes approach are clear since when combined with a suitable turbulence model these equations represent the flow and heat transfer physics discussed previously. In particular, the Navier-Stokes equations accurately represent possible separated regions and regions of significant secondary flow. In addition, the Navier-Stokes approach allows representation of the entire flow field by a single set of equations thus avoiding problems associated with representing different regions of the flow by different equations and then matching flow regions. Although turbulence modelling remains a problem, turbulence modelling presents the same difficulties with alternate approaches, and alternate approaches cannot necessarily represent the highly complex flow physics to be found in general turbine stator flow fields.

However, application of the Navier-Stokes equations to the turbine stator problem is not an easy task. The presence of complex geometries, rapidly varying flows, strong secondary flows and the need for adequately resolving the various physical scales are items which all present their own problems. Nevertheless, with continued improvement of numerical solution techniques the Navier-Stokes approach is becoming a practical procedure for the three-dimensional turbine stator problem.

APPROACH

The present approach solves the ensemble-averaged Navier-Stokes equations via the Linearized Block Implicit (LBI) of Briley and McDonald (Ref. 1). Boundary conditions for subsonic inflow and outflow (the usual case) set upstream stagnation pressure, upstream stagnation temperature, upstream flow angle, and downstream static pressure. Additional conditions used are density derivative on the inflow (upstream boundary), and velocity and temperature second derivatives on

*Work done under NASA Contract NAS3-24358.

the downstream boundary. On the cascade blade no-slip conditions and a zero pressure gradient condition are applied along with either a specified temperature or a specified heat transfer rate. In applying the no-slip wall conditions, proper wall resolution is mandatory. In general, the first grid point off the wall is taken so as to place a point in the viscous sublayer. The governing equations are written in general tensor form and solved in a body-fitted coordinate system. Details of the governing equations, numerical technique, grid construction, turbulence model, etc. are given in Refs. 2-4.

Using this approach in a previously described HOST effort (Refs. 3 and 4), two- and three-dimensional calculations were made for a turbine cascade. The purpose of the first effort was two-fold; (i) demonstration of the Navier-Stokes current capability in simulating these complex flow fields, and (ii) demonstration that the Navier-Stokes approach is a practical procedure with current technology. The first goal was reached via heat transfer and pressure distribution comparisons with experimental data for two-dimensional cases and a demonstration three-dimensional laminar cascade calculation. The quantitative comparisons for the two-dimensional cases showed good agreement between calculation and measurement; a comparison between predicted and measured heat transfer on a C3X airfoil with specified transition location is shown in Fig. 1. In addition, the demonstration three-dimensional calculation showed many of the expected features such as the saddle point singularity in the endwall boundary layer as shown in Fig. 2.

In addition to providing accurate simulations, the Navier-Stokes approach, if it is to be practical, must have reasonable run times to convergence. Long run times, even if leading to accurate simulations, would severely hamper application of the Navier-Stokes procedure to practical turbine stator or rotor problems on a regular basis. Computer run time is based upon run time per time step and number of time steps to convergence. In regard to the first item, the current Navier-Stokes cascade code, BMINT/CX, requires approximately 1.2×10^{-4} secs/grid point/time steps on a CRAY-1 computer which for typical two-dimensional cases having 3500 grid points gives 0.4 secs/time step. A three-dimensional case of 70,000 grid points, due to additional code overhead, would require approximately 12 secs per time step. In regard to time steps to convergence selected cases showed essential convergence for engineering purpose in 70-100 time steps whereas other cases required approximately 150 time steps convergence. Both of these cases are within the constraints required for the Navier-Stokes approach to be a practical approach to the turbine blade row problem.

PRESENT EFFORTS

The present turbine cascade program focuses upon three efforts: (i) further code development with particular emphasis on increasing the number of grid points which can be run on a given machine, (ii) further convergence studies, and (iii) calculation of the turbulent flow through a three-dimensional turbine stator. In regard to the first objective, this will be passive to the user and will simply allow for a higher resolution calculation by utilizing out-of-core storage. The second objective would detail the convergence properties of the procedure. Results obtained in Refs. 3 and 4 indicate rapid convergence insofar as engineering predictions are concerned. Under the present effort these convergence properties are considered in more detail for a wider variety of cases.

The initial test case chosen for the convergence study was the Turner turbine cascade (Ref. 5). The convergence study calculation was run with a 'C' grid containing 113 pseudo-azimuthal grid points, and 30 pseudo-radial grid points. High wall resolution was obtained with the first point off the wall being approximately 1.5×10^{-5} chords from the blade surface. In regard to convergence, several criteria can be considered. These include surface pressure distribution, maximum normalized residual and pressure coefficient at the stagnation point. In regard to these factors, it should be noted that in the present calculations it is the converged steady state flow field which is the item of interest. Therefore, although the current numerical procedure solves the unsteady flow equations, it is not necessary and, in fact, is uneconomical to obtain a time-accurate solution when seeking steady state solutions. Instead, matrix preconditioning techniques are used to obtain a converged solution as rapidly as possible. In these studies, the calculation was initiated from a very simple flow field in which the velocity magnitude and static pressure were set constant throughout the flow with the velocity flow angle a function of axial location. Very simple profiles were used near the blade surface to bring the velocity to the no-slip condition.

A plot showing the maximum normalized flow residual is presented in Fig. 3. As can be seen, the maximum residual drops slightly over 4 orders of magnitude in 150 time step iterations and then levels. In general, based upon previous experience, three orders of magnitude drop in residual give convergence suitable for many engineering applications. However, in addition to monitoring the residual behavior, it is necessary to consider the flow field dependent variable behavior. Based upon experience one sensitive item is the pressure coefficient at the stagnation point where C_p is taken as $(p - p_{ref})/1/2\ \rho_{ref}\ q^2_{ref}$. The reference quantities are taken from the inflow boundary and consequently since only total pressure, total temperature and flow angle are specified, these may vary with time. The results show the variation of stagnation point C_p with time-step iteration number. The calculation was run with an inflow Mach number of approximately 0.24; on an inviscid basis this should lead to a stagnation point C_p of approximately 1.015. The present results converge to a value of approximately 1.005 for the Navier-Stokes simulation. As can be seen in Figs. 4 and 5, the stagnation point C_p was obstensibly converged at 100 time steps although some slight oscilations occurred until 150 time steps. These results clearly indicate the rapid convergence of the present Navier-Stokes procedure and its potential for use as an engineering tool. It is expected that this type of convergence shall also be obtained for three-dimensional calculations.

In addition to these results, the present effort is aiming at the three-dimensional cascade problem for turbulent flow. A calculation shall be done for a modern turbine cascade design, the convergence properties of the procedure assessed and if possible, a comparison made between computed flow field values and values obtained by experimental measurement.

REFERENCES

1. Briley, W.R. and McDonald, H.: Solution of the Multidimensional Compressible Navier-Stokes Equations by a Generalized Implicit Method. Journal of Computational Physics, Vol. 24, pp. 372-397, 1977.

2. Shamroth, S.J., McDonald, H. and Briley, W.R.: Prediction of Cascade Flow Fields Using the Averaged Navier-Stokes Equations. ASME Journal of Engineering for Gas Turbines and Power, Vol. 196, pp. 383-390, 1984.

3. Yang, R.-J., Weinberg, B.C., Shamroth, S.J. and McDonald, H.: Numerical Solutions of the Navier-Stokes Equations for Two- and Three-Dimensional Turbine Cascades with Heat Transfer. NASA CR-174929. 1985.

4. Weinberg, B.C., Yang, R.-J., McDonald, H. and Shamroth, S.J.: Calculation of Two- and Three-Dimensional Transonic Cascade Flow Fields Using the Navier-Stokes Equations. ASME Paper 85-GT-66, 1985.

5. Turner, A.B.: Local Heat Transfer Measurements on a Gas Turbine Blade. Journal of Mechanical Engineering Sciences, Vol. 13, 1971.

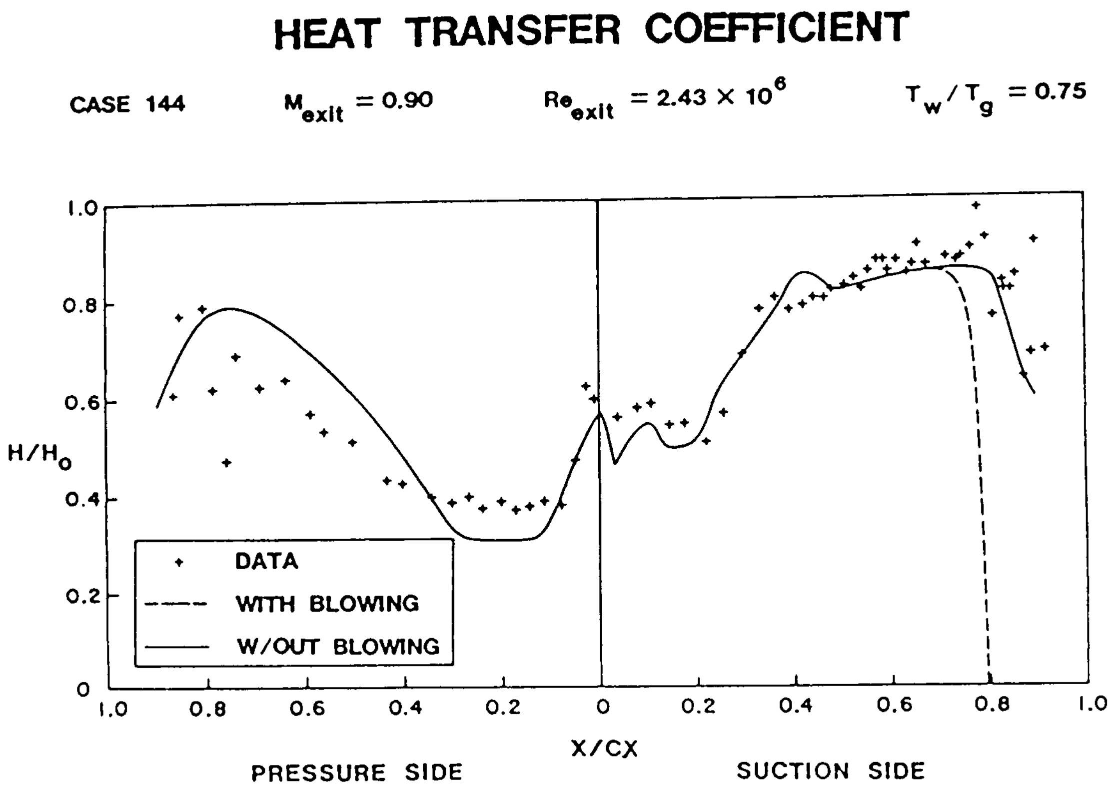
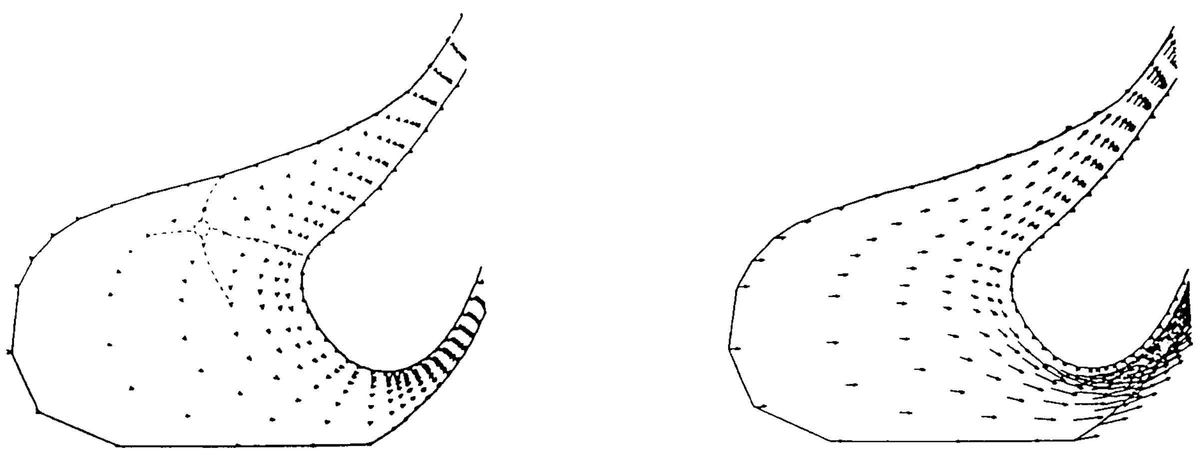


FIGURE 1



(a) VECTOR PLOT ON 0.135% SPANWISE PLANE **(b) VECTOR PLOT ON MIDSPAN PLANE**

FIGURE 2

CONVERGENCE RATE

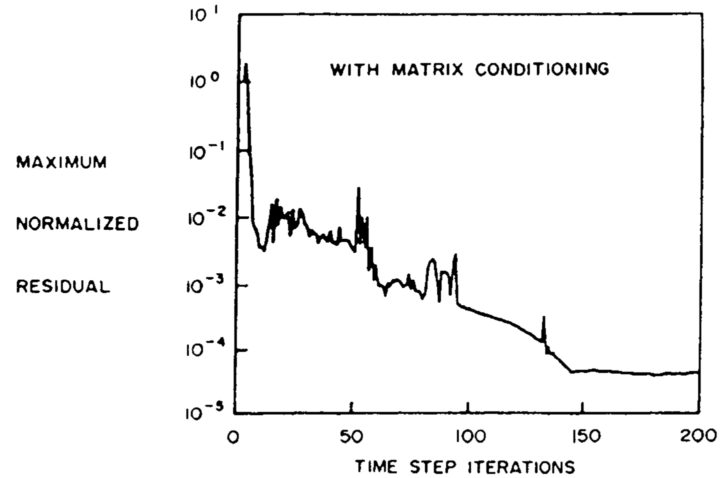


FIGURE 3

CONVERGENCE OF PRESSURE AT STAGNATION POINT

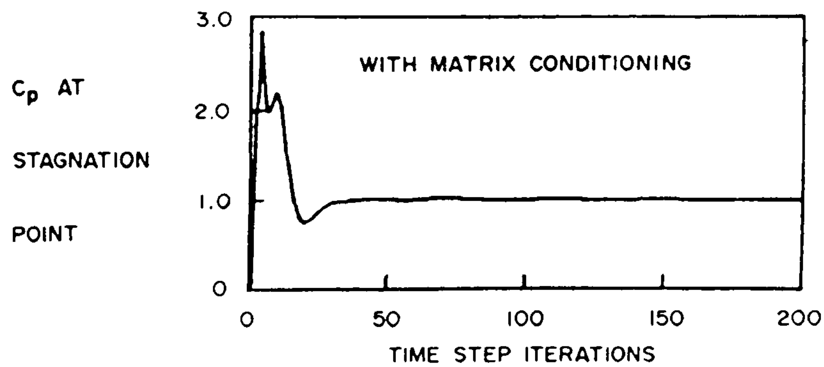


FIGURE 4

CONVERGENCE OF PRESSURE AT STAGNATION POINT

REDUCED SCALE PLOT

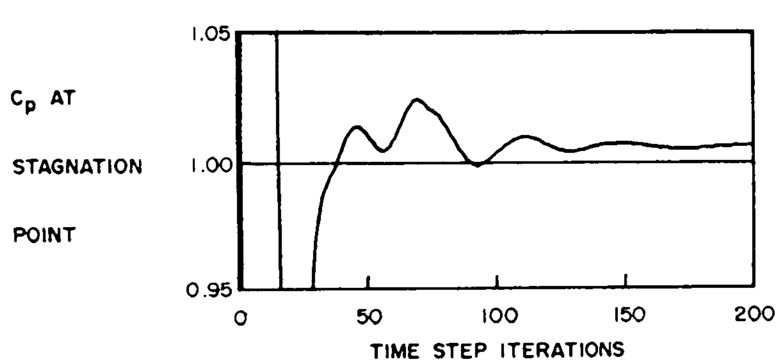


FIGURE 5